Vitronics Soltec

By G. Schouten

Skipped measuring solderpads in wave soldering

Introduction

Measuring pads have normally no mounting hole or outgassing hole in the pad. If the pad aperture dimension in the solderresist is relatively small, there is a risk that the solderwave skips that pad.
That risk increases if the solderresist layer is thicker than 35 microns and if the aperture has a relatively sharp edge.
Finally if the solderability of such a pad is not optimal, skipped joints will be very common in that case.
This sheet will explain the cause(s)

Skipped pads

If PCBs are covered with a relative thick solderresist layer, with sharp edges at the apertures, there is a risk of skipped pads after a wave soldering process.
Since solderresist is a non-solderable material the solder is not able to wet it. If liquid solder makes contact with a non-wettable material the solder tries to recede and forms a wetting angle of 180°. So the solder will not easily spread over such a surface. In the practical situation this means that if the solder comes at an edge at a non-wettable surfaces it will not spontaneously follow the contour of surface over the edge. As a matter of fact it will have a tendency to withdraw from such an edge that is connecting non-wettable surfaces. This means that the solder contacts the metal parts beyond such an 'edge barrier' just a few tenths of a millimetre after this edge is passed.

The 'gap' between the solderresist and the solderable pattern that is created by this effect is in most cases filled with flux, since this flux is not washed away by the solderwave as the board passes over the wave. This entrapped flux will in contact with the solderwave evaporate very fast.
This creates a vapour pressure inside the solderpad-resist aperture. Due to this vapour pressure the solder is not able to contact the pad inside the aperture, resulting in a skipped pad.

Using a chipwave might help, since this wave will disturb the flux vapour pressure build-up in the aperture.

There are however cases that will hinder proper pad wetting even more.
It is known that copper tracks, coated with OSP as solderability protection are in general more vulnerable for this effect.
The reason for this is that the OSP coating must first be dissolved by the flux (solvent), before the flux really can contact the metal parts to be wetted by the solder.
If the coating was applied some time ago, the molecules of the coating will be harder to break by the solvent, due to the fact that these molecules will polymerise during storage. These molecules become longer and therefore more difficult to dissolve.
Due to this effect the flux needs more time to get to the metal underneath this OSP coating.
If during this process the vapour of the entrapped flux will displace the solderwave from the pad surface, solderwetting becomes impossible.

Information Sheet 098

Another drawback of OSP that has been too long on the PCB, is that this layer will form small cracks in due time. Via these cracks the oxygen will now face the metal and will create oxides.
Next these extra oxides have to be removed by the flux also.

Sometimes the choice of a flux with an appropriate solvent can give a solution for this problem, but in general a dynamic wave like the chipwave might give the best support to minimise these skipped pads.

Vitronics Soltec

DISCLAIMER
All content is subject to periodic review and may be changed without notice.
Vitronics Soltec BV assumes no obligation for content contained herein.

COPYRIGHT VITRONICS SOLTEC BV
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, without the prior written permission of Vitronics Soltec BV.
This publication remains the property of Vitronics Soltec BV and may not be passed, loaned or given to any third party.

Vitronics Soltec BV reserves the right to make changes in design and specifications without notice.